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This section will cover the basic use of the Part Design workbench to create parts. This section will consist of three parts: basic shapes, operations on shapes and interfacing between part design and sketcher. Basic Shapes Part Design & Sketcher - Engineering CATIA Part Design & Sketcher CATIA® V5R19 Part Design & Sketcher CATIA Version 5 uses the Sketcher workbench as its principal method to create profiles. These profiles can be constrained using many different types of constraints. The first objective of the course is to learn to use the Sketcher and constrain your profiles to the desired specifications. CATIA Part Design & Sketcher - cadcamlab.org In Analysis tab of action bar, click on sketch analysis icon. A panel will appear, it will tell you. if the sketch is constrained, under constrained or over constrained; if it is opened or closed; the list of all the elements in your sketch (points, lines, arc, splines, etc.) Since the sketch is opened, click on close opened profile feature! Warning ! this will close the profile but also change the geometry of your sketch 7-tips Sketcher Part Design CATIA R19x | GrabCAD Tutorials CATIA Part Design & Sketcher

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to Draw a Spur ...The Part Design application makes it possible to design precise 3D mechanical parts with an intuitive and flexible user interface, from sketching in an assembly context to iterative detailed design. Part Design application will enable you to accommodate design requirements for parts of various complexities, from simple to advance. Part Design 1 « CATIA Tutor Two simple ways are by using the top pull down menu (Start – Mechanical Design – Sketcher), or by selecting the Sketcher icon. When you enter the sketcher, CATIA requires that you choose a plane to sketch on. You can choose this plane either before or after you select the Sketcher icon. To exit the sketcher, select the Exit Workbench icon. CATIA V5 Basic Tutorial | GrabCAD Tutorial This video describes the initial step of part modeling & sketch positioning. CATIA V5 SKETCH POSITIONING - YouTube CATIA Part Design & Sketcher 3DEXPERIENCE® R2017x Part Design & Sketcher CATIA Version 6 uses the Sketcher workbench as its principal method to create profiles. These profiles can be shaped and located via many different types of constraints. The first

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the major features of CATIA and directs you to CATIA's online documentation for a more detailed description of the commands when appropriate. The workbenches covered in this book are; Sketcher, Part Design, Assembly Design, Drafting, Generative Surface Design, and Imagine and Shape. Preceding each tutorial is a brief

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